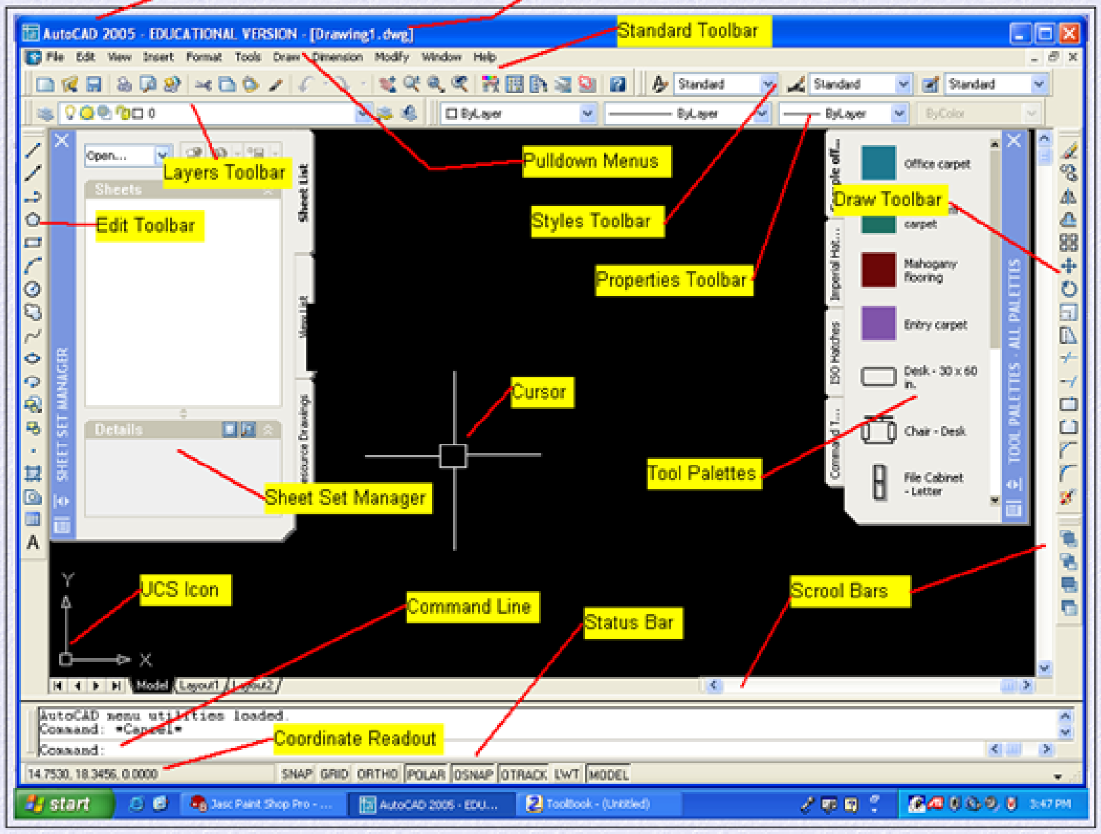
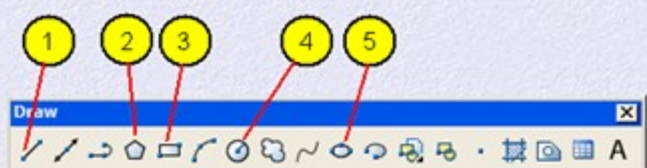
AutoCAD Basics

AutoCAD Interface

AutoCAD Version Number - Shows what AutoCAD version you are running.
Current open drawing name - Shows the name of the drawing you currently have open.
Standard Toolbar- Toolbar for standard AutoCAD commands.
Pulldown Menus - Easy access to AutoCAD commands.
Layers Toolbar - Access AutoCAD layer commands.
Edit Toolbar - Access to AutoCAD editing commands.
Styles Toolbar - Access AutoCAD styles toolbar.
Properties Toolbar - Access properties toolbar
Cursor - Used to point to objects in AutoCAD.
Draw Toolbar - Access to AutoCAD draw commands.
Tool Palettes - Access to tool palettes.
Scroll Bars - Used to scroll around drawing in drawing area.
Status Bar - Used to turn on and off AutoCAD settings.
Command Line - Used to type in AutoCAD commands from keyboard.
Coordinate Readout - Used to keep track of cursor location in drawing area.
Sheet Set Manager - Access sheet set manager.



Draw Commands



Draw Toolbar used to access AutoCAD draw commands you can also use draw pulldown menu or type the command in at the Command line. The draw commands create objects such as lines, circles, and ellipse's. An object is the smallest component of a drawing. A drawing is made up of a combination of these objects. When creating objects with draw commands, AutoCAD always prompts "Ask" you to indicate points (such as endpoints, centers, or radii) to describe the size and location of the object to be drawn. Drawing commands can be entered from the keyboard (command Line), the Draw Toolbar, or the Draw Pull-down menu.

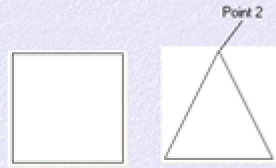
1. **Line** - Draw lines in AutoCAD
2. **Polygon** - Draws polygons with three or more sides
3. **Rectang** - Used to draw rectangles
4. **Circle** - Command used to draw circles
5. **Ellipse** - Use to draw ellipse

Line Command

Line Command use to draw lines in AutoCAD

1. Press **F8** on keyboard to turn on **ORTHO**
2. Click on the **LINE** icon in the draw toolbar.
3. Specify first point: **Pick any point on your screen with the mouse**
4. Move your cursor to the right a little do not click down.
5. Type in: **2 (press enter)**
6. Move your cursor up a little do not click down
7. Type in: **2 (press enter)**
8. Move your cursor to the left a little do not pick down
9. Type in: **2 (press enter)**
10. Move you cursor down a little do not click down
11. Type in: **2 (press enter)**
    you should have drawn a perfect box see figure 1
12. **Press the ESC** key in the upper left corner of your keyboard to cancel the **LINE** command.

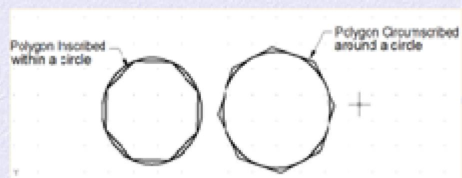
1. Click on the **LINE** icon
2. Specify first point: **Pick any point on your screen with the mouse**
3. Move your cursor to the right a little do not click down
4. Type in: **2 (press enter)**
5. Press **F8** on the keyboard to turn off **ORTHO**
6. Specify next point: **Pick point 2 with your mouse**
7. Type in : **C (press enter)**
   you should created something that resembles a triangle see image.



Polygon Command

Command Use to draw a polygon with three or more sides

1. **Draw two circles any diameter using the circle icon.** (Refer to the circle command if you do not know how to use this command
2. Click on the **POLYGON** icon in the draw toolbar
3. Enter number of sides: at this point you can enter as many sides as you wish. (No less than three sides). At this time type in : **8 (press enter)**
4. Specify center of polygon type in: **CEN (press enter)**
5. Move your cursor around the outer edge of one of the circles
   until you see a **small yellow circle** at the center of the bigger circle. Hold the cursor there until the word **CENTER** appears then click down with the mouse.
6. Enter the option type, type in: **C (press enter)**
7. Specify radius of circle type in: NEAR (Press enter)
8. Move the cursor to outer edge of the circle hold it there until you see a **small yellow hour glass** and the word **NEAREST** appears then click down with the mouse.
   You may put a polygon within a circle (Inscribed) by typing in, in step 6 an "I" instead of a "C"



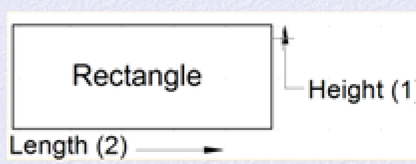
Rectangle Command

Command used to draw a rectangle

1. Click on the **RECTANGLE** icon in the draw toolbar.
2. Specify first corner point: **click anywheres on your screen**
3. Specify other corner: **move your cursor up and to the right any distance you wish then click down.**

If you wish to Draw a rectangle a specific size do the following:

1. **Repeat steps 1 and 2**
2. Specify other corner type in: **@3,1 (press enter)**
   Using this option you have entered a specific length and height for your polygon. You may change the numbers to anything you wish but leave the @ and the , in its exact location. Broken down the 3 is the length and the 1 is the height. The @ symbol tells AutoCAD you are using relative co-ordinate entry.



Circle Command

Command used to draw circles

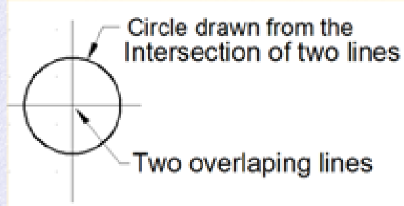
1. Click on the **CIRCLE** icon in the draw toolbar.
2. Specify center point of circle: **click anywheres in the drawing area.**
3. Specify radius of circle Type in: **.25 (press enter)**

To draw a circle by specifying a circle diameter do the following:

1. **Repeat steps 1 and 2.**
2. Specify radius of circle Type in: **D (press enter)** The **D stands for diameter**
3. Specify diameter of circle Type in: **1 (press enter)**

To draw a circle at the intersection of two lines:

1. Draw two lines that overlap using the **line command**
2. Click on the **CIRCLE** in the draw toolbar
3. Specify center point of circle Type in: **INT (press enter)**
4. Move your cursor to the intersection of the two lines until you see a **small yellow X** and the word **INTERSECTION** appears. Then click down.
5. Now you can either **type in a radius or type in D for diameter** and move on the next prompt to type in the diameter.



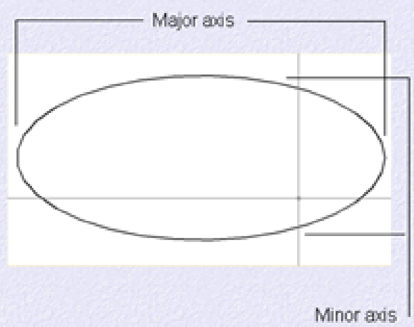
Ellipse Command

Command use to draw ellipses

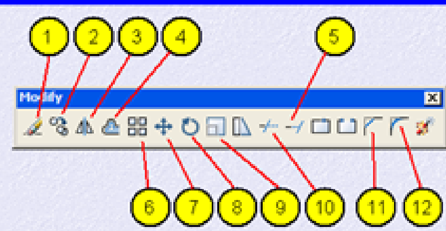
1. Click on the **ELLIPSE** icon in the draw toolbar.
2. Specify axis endpoint: **Click anywheres on your screen**
3. Specify other endpoint: **Pick another point a distance you specify to the right of the first point you picked**. This will be the major axis of your ellipse.
4. Move your cursor up a distance you specify and click. This will be your minor axis.

To draw an ellipse with a center, rotation, and angle do the following:

1. **Repeat step 1 above.**
2. Specify axis endpoint of ellipse Type in: **C (press enter)**
3. Specify center of ellipse: **Pick anywheres in the drawing area**
4. Specify the endpoint of axis, move your cursor to the right a little then type in: **.50 (press enter).** You have entered the radius of the circle you are trying to make an ellipse from. The radius of the circle is all that is needed.
5. Specify distance to other axis Type in: **R (press enter)**
6. Specify rotation around major axis Type in: **30 (press enter)** You have entered the rotation angle of the line of sight of the ellipse. For example if you were looking at an ellipse at a 30 degree angle, the rotation would be 30 degrees.



Modify Toolbar



Modify Toolbar used to access AutoCAD draw commands can also use Modify pulldown menu or type the command in at the Command line. After you have created some objects in AutoCAD, objects that make up a technical drawing, such as lines and circles or a combination of both, they are sometimes just not the way you would like them, they may need to be rotated, moved, copied, scaled larger or smaller, etc. To do these things you need to have a good basic understanding of some of the AutoCAD editing commands. In this module are 12 of the most basic AutoCAD editing commands. With these 12 commands you can do 90 percent of most editing operations. Practice these 12 commands learn them well. The best way to access these commands is by using the editing toolbar at the right of the AutoCAD opening screen.

1. **Erase** - Erase object in the drawing area.
2. **Copy** - Used to copy one or more objects.
3. **Mirror** - Command used to mirror an exact duplicate of an object.
4. **Offset** - Used to offset one object from another a distance you specify.
5. **Extend** - Used to extend one line to another
6. **Array** - Use to make a rectangular or polar array of an object.
7. **Move** - Used to move objects around in the drawing area.
8. **Rotate** - Used to rotate an object around a base point.
9. **Scale** - Command used to make an object larger or smaller.
10. **Trim** - Command used to trim an object from another object.
11. **Chamfer** - Used to put a chamfer between two lines.
12. **Fillet** - Use to put a fillet between two lines a radius you specify.

Erase Command

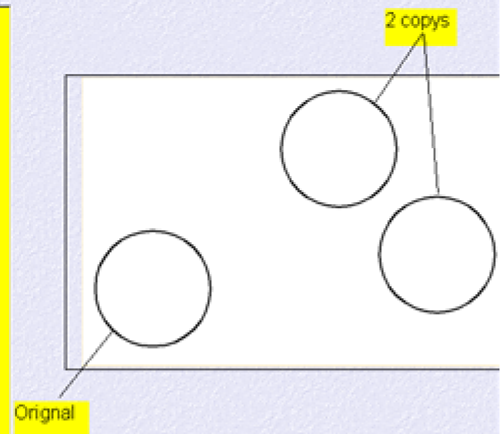
Command used to erase objects in the drawing area

1. Draw a line and a circle any length any diameter.
2. Click on the **ERASE** icon in the edit toolbar.
3. Select objects: **Select all objects that you would like to erase with the pickbox. When you are done selecting objects press ENTER on the keyboard. The objects should disappear.**

Copy Command

Command used to copy objects in the drawing area

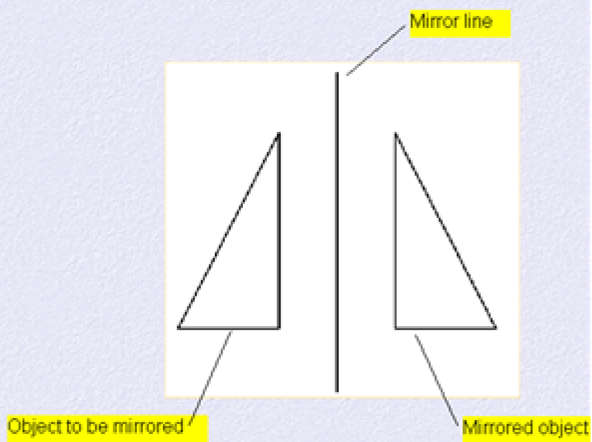
1. Draw a circle any diameter.
2. Click on the **COPY** icon in the edit toolbar.
3. Select objects: **Select the circle with the pickbox (on the line) when the circle is highlighted press the ENTER key on the keyboard.**
4. Select objects: 1 found **(This line tells that you have selected only 1 object)** Select objects: **(This line ask you if you would like to select more objects if not press ENTER on the keyboard).**
5. Specify base point of displacement: **Pick with the cursor near the center of the circle.**
6. Move the object to the location you desire and click down with the mouse.



Mirror Command

Command used to mirror an object to the side of another object

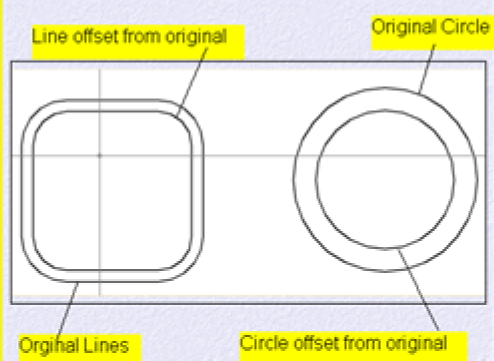
1. Draw a rectangle any size using the rectangle command.
2. Click on the **MIRROR** icon in the edit toolbar.
3. Select objects: **Select the rectangle with the pickbox (on the line)**
4. Select objects: press **ENTER** on the keyboard.
5. Specify point on mirror line: **Move your cursor to the right and above a short distance from the rectangle. Click down with the mouse.**
6. Specify first point of mirror line: **Specify second point of mirror line: move your cursor down a short distance then click down with the mouse.**
7. Delete source objects Type in: **N** (press enter). An exact duplicate of the rectangle you created in step 2 will be created to the right. **Note: If you create a mirror line above the rectangle an exact duplicate of the rectangle will be created above. You can mirror any object in the drawing area text, lines, circles, views, etc.**



Offset Command

Command used to offset one object from another

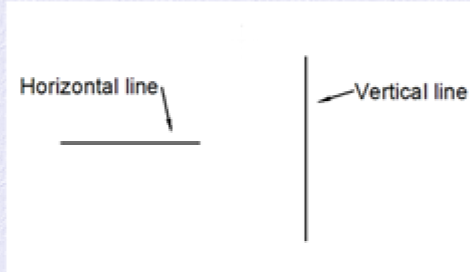
1. Draw a circle and a line any diameter any length.
2. Click on the **OFFSET** icon in the edit toolbar.
3. Specify offset distance Type in: **.50** (press enter)
4. Select object to offset: **Select the line with the pickbox.**
5. Specify point on side to offset: **Pick just above the line you have selected in step 4 with the cursor. An exact duplicate of the first line you selected in step 4 is created just above .50 distance away. If you would have picked below the line you picked in step 4 an exact duplicate would be created .50 below.**
6. Select object to offset: press **ENTER** on the keyboard. **You can offset almost any object in the drawing area. You can offset text. You can also offset a circle to the inside or outside itself. To offset inside a circle click inside the circle. To offset outside the circle click outside the circle.**



Extend Command

Command used to extend one line to another

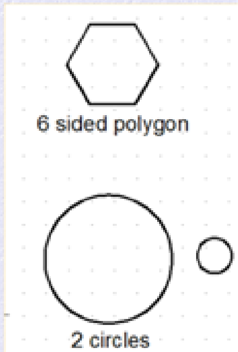
1. Draw two lines one horizontal and one vertical
   Draw the vertical line a short distance
   away from the horizontal line. (See image)
2. Click on the **EXTEND** icon in the edit toolbar.
3. Select boundary edges........
   Select objects: **select the vertical line (the line you want to extend to).**
4. Press **ENTER** on the keyboard
5. Select objects to extend: **Select the right end point of the horizontal line with the pick box.**
   The horizontal line will now extend to the vertical line.
6. Press the **ESC** key on the keyboard to cancel the command.



Array Command

How to do a rectangular array:
1. Start a new drawing from scratch.
2. Draw a polygon, **6 sides, inscribed, with a radius of .50.**
3. Click on the **ARRAY** icon in the edit toolbar.
   The array dialog box opens.
4. Set **ROWS to 6**, set **COLUMNS to 6.**
5. Set **ROW OFFSET to 1.50**, set **COLUMNS OFFSET to 1.50.**
6. Click on **SELECT OBJECTS** icon.
7. Select the polygon with the pickbox (on the line).
   Then press **ENTER** on the keyboard.
8. Click on **OK**.
   Note: The row and offset distance is taken from the center of the polygon.

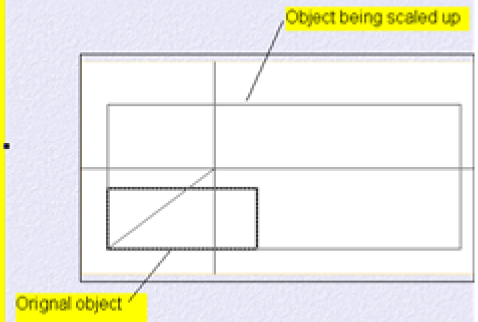
How to do a polar array:
1. Draw two circles diameter (see image)
2. Click on the **ARRAY** icon.
   The array dialog box opens
3. Click on **POLAR ARRAY**.
4. Click on **SELECT OBJECTS** icon. **Click on the small circle (on the line) then press ENTER on the keyboard.**
5. Click on the **PICK CENTER POINT** icon Type in: CEN (press enter). Move the cursor over to the large circle (on the line). When a small yellow circle appears at its center click down with the mouse.
6. Set **TOTAL NUMBER OF ITEMS to 6**.
   Set **ANGLE TO FILL to 360**
7. Click on **OK**. 6 small circles should appear around the larger circle.



Scale Command

Command use to scale an object larger or smaller in size

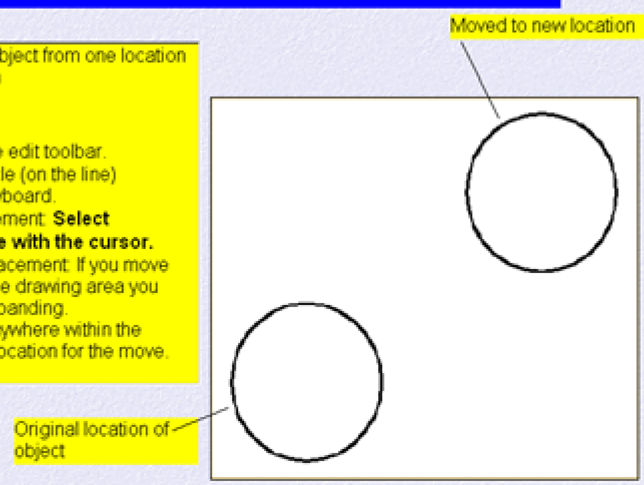
1. Draw a circle any diameter
2. Click on the **SCALE** icon in the edit toolbar.
3. Select objects: Select the circle (on the line) then press **ENTER** on the keyboard.
4. Specify base point: **Click near the center of the circle with the cursor.**
5. Specify scale factor or reference: Type in: **2** press enter on the keyboard. This makes the circle 2 times its original size. If you were to type in .50 it would make the circle half its original size. This can be done with any object created in the AutoCAD drawing area a percentage less than 1 makes the object smaller. A percentage more than 1 makes the object bigger. A percentage of 1 has know effect on the object.



Move Command

Command used to move an object from one location to another in the drawing area

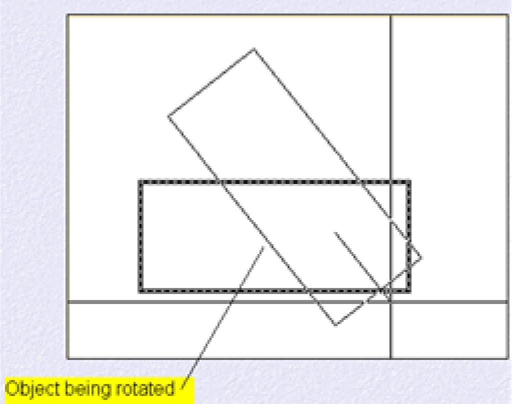
1. Draw a circle any diameter
2. Click on the **MOVE** icon in the edit toolbar.
3. Select objects: Select the circle (on the line) then press **ENTER** on the keyboard.
4. Specify basepoint of displacement: **Select near the center of the circle with the cursor.**
5. Specify second point of displacement: If you move your cursor around a little in the drawing area you can see what is called rubberbanding.
6. Click down with the mouse anywhere within the drawing area at the desired location for the move.



Rotate Command

Command used to rotate an object around a point you pick

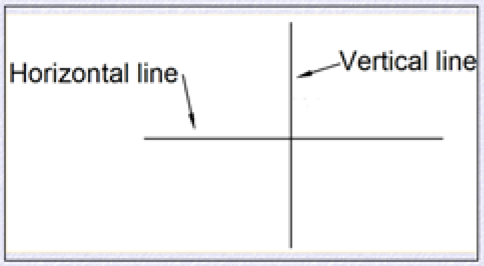
1. Draw a rectangle with only two equal sides using the rectangle command.
2. Click on the **ROTATE** icon in the edit toolbar.
3. Select objects: Select the rectangle (on the line) then press **ENTER** on the keyboard.
4. Specify base point: **Select somewheres near the center of the rectangle with the cursor.**
5. Specify rotation angle: **At this point you can type in an angle of rotation at the command line or by moving you mouse around you can dynamically see the desired rotation angle, then click down with the mouse when you like the rotation angle.**



Trim Command

Command used to trim one line back from another line

1. Draw two overlapping line one horizontal and one vertical. (See image)
3. Click on the **TRIM** icon from the edit toolbar.
4. Select cutting edges..... Select objects: **Select the Vertical line with the pick box (this will be your cutting edge).**
5. Press **ENTER** on the keyboard.
6. Select object to trim: **Select the horizontal line the part to the right of the vertical line (this is your object to trim).**
7. Press the **ESC** key on the keyboard to exit the command.

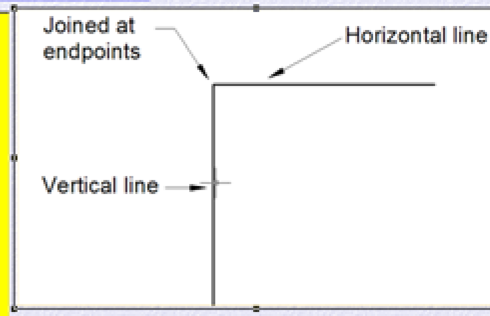


Chamfer Command

Command use to put a chamfer between two connecting lines

1. Draw a 1" horizontal line and a 1" vertical line joined at endpoints. (See image)
2. Click on the **CHAMFER** icon in the edit toolbar.
3. Select first line Type in: **D** (press enter) This is to set the distance of the chamfer.
4. Specify first chamfer distance Type in: **.25** (press enter)
6. Specify second chamfer distance Type in: **.25** (press enter)
4. Select first line: **Select the horizontal line somewheres close to the endpoint near the vertical line (But not on the endpoint itself) with the pickbox.**
5. Select second line: **Select the vertical line somewheres close to the endpoint near the horizontal line (but do not select the endpoint itself) with pickbox. You should now have a .25 chamfer at 45 degrees.**

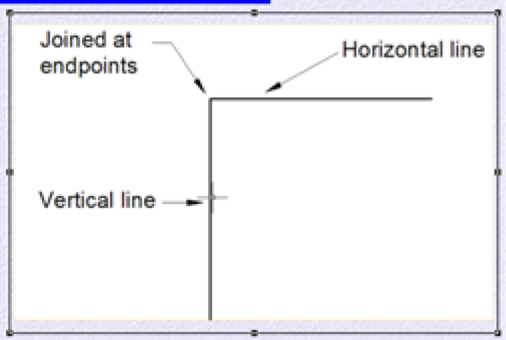
Note: When both chamfer distances are the same the chamfer angle will always be at 45 degrees. If you would like a chamfer at an angle other than 45 degrees just make the two distance settings a different value.



Fillet Command

Command used to put a fillet between two lines

1. Draw a 1" horizontal line and a 1" vertical line joined at endpoints. (See image)
2. Click on the **FILLET** icon in the edit toolbar.
3. Select first object Type in: **R** (press enter) This is to set the fillet radius.
4. Specify fillet radius Type in: **.25** (press enter)
5. Select first object: **Select the Horizontal line somewheres near the endpoint close to the vertical line (do not click on the endpoint itself) with the pick box.**
6. Select second object: **Select the vertical line somewheres near the endpoint close to the horizontal line (do not pick on the endpoint itself) with the pick box. There now should be a fillet with a radius of .25 between the two lines.**



Coordinate Entry

A good understanding of how co-ordinates work in AutoCAD is absolutely crucial if you are to make the best use of the program. It is an exact way of entering precise points and location within the AutoCAD drawing area, such as the starting and ending points of lines, exact centers of circles and so on. The co-ordiante entry system use in AutoCAD is called the " Cartesian Coordinate System".

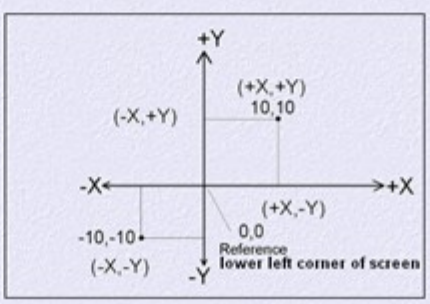
X,Y,Z Co-ordinate Entry System
Angular Measurement

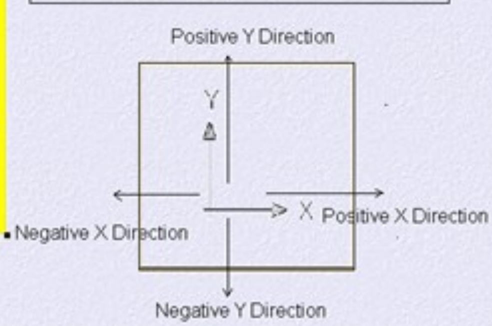
Four most popular way of enter coordinates in autocad:

Absolute Coordinate Entry
Relative Coordinate Entry
Polar Coordinate Entry
Direct Distance Entry

X,Y,Z Co-ordinate Entry System

A good understanding of how co-ordinates work in AutoCAD is absolutely crucial if you are to make the best use of the program. The co-ordinate system in AutoCAD is called the Cartesian co-ordinate system. The position of a point can be described by its distance form two axes, X axes and Y axes. The UCS icon in the lower left corner of the drawing area shows you which way the X and Y axes go. In AutoCAD when you enter a coordinate using either absolute or relative coordinate entry, AutoCAD always reads the X axes first and then the Y axes second . Every thing you draw in AutoCAD lines, circle, etc. always has an X and Y coordinate location assigned to it. The coordinate readout at the bottom left corner of the screen always tells you where your cursor is at in XY. If you move your cursor around the XY coordinate readout will change. The lower left corner of your screen is the 0 reference point in the drawing area, this is the origin where the coordinate readout begins counting from. You must understand which way is Y and Which way is X and where the 0 reference point is. The 0 reference point is at the exact intersection of the X and Y on the UCS icon.

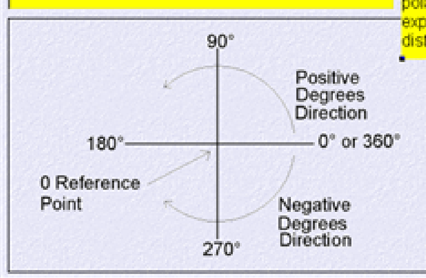




Coordinate Entry

AutoCAD by default measure's angles (degrees) counter -clockwise as a positive angle starting from 0 to 360 degrees. If an angle is measured in the clockwise direction this is considered a negative angle starting from 360 to 0. In AutoCAD 0 degrees is the same as 360 degrees, they occupy the same point. 0 or 360 degrees is going to the right, 90 degrees is going straight up, 180 degrees is going to the left, 270 degrees is going straight down. All other angles lie in between the four major angles.

Drawing a line in the positive degrees direction in AutoCAD is easy you just tell AutoCAD how far and at what angle you want to draw the line. (Example: (@6<150) draws a line 6 inches long in the positive 150 degrees direction. Drawing a line in the negative direction would look like this (Example: (@4<-30) draws a line 4 inches in the -30 direction. (Refer to polar coordinate entry for a better explanation on how to draw lines at distances and angles

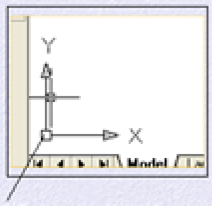


Absolute Coordinate Entry

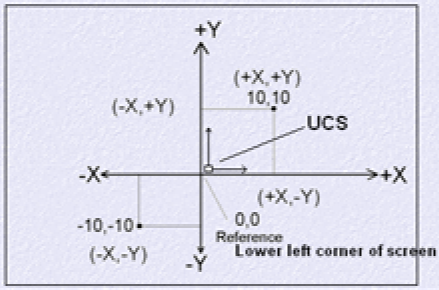
Using this method, you enter the points as they relate to the origin of the WCS (World Co-ordinate System). The origin of the WCS is at the lower left corner of your drawing area, where the UCS icon is. For example you would like to start a line at 10,10 in the drawing area that's 10 on the X axes and 10 on the Y axes. You enter the line command, AutoCAD prompts you to specify first point, you type in: 10,10 and press enter. The line commands puts the first point of the line at 10,10 from the 0 reference point in the drawing area, from that point you can enter another point for the second point of the line. To start the first point of the line in the -X and -Y direction, you would enter the line command for first point type in: -10,-10 and press enter. To put it simply all coordinates using the absolute method of entry come relative from the 0 reference point at the bottom left corner of your screen. **(Click here to do a short tutorial)**

Example's of absolute coordinate entry would be 2,2 , -2,1 , 1,-2 , -2,-2

Remember: AutoCAD always reads X axes first then the Y axes.



0,0 Reference Point (orgin)



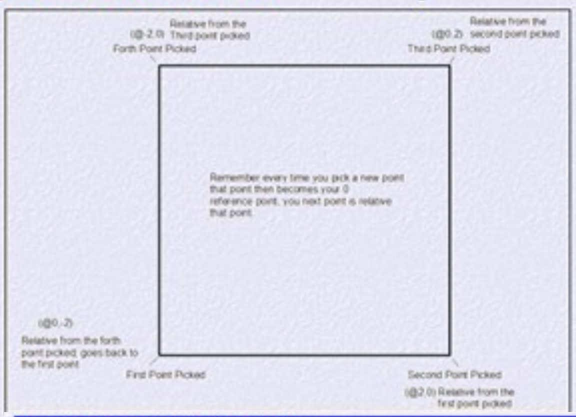
Absolute Coordinate Entry

1. Type in: **LINE** (press enter)
2. Specify first point Type in: **2,2** (press enter) starting point.
3. Specify next point type in: **4,2** (press enter) draws a line to second point
4. Specify next point type in: **4,4** (press enter) draws a line to the third point
5. Type in: **CLOSE** (press enter) Closes the third point to the first point with a line.

Remember: AutoCAD always reads the X coordinate first then the Y coordinate.

Relative Coordinate Entry

The relative coordinate entry method allows you to enter points in reference to the last point picked. You enter the line command and pick the starting point of a line that point then becomes the 0 reference point. You are now prompted to enter the second point of your line, You enter @2,2 AutoCAD then draws a line relative from the last point you picked to the second point. You are now prompted to enter another point, you type in @3,3 AutoCAD now draws a line relative form the second point to the third point. The @ symbol tells Autocad that you are going to use the relative coordinate entry to locate another point, this symbol must always be put in front of the X and Y coordinate. **(Example's of relative entry: @2,2 , @1,2 , @6,5)**
(Click here to do a short tutorial)



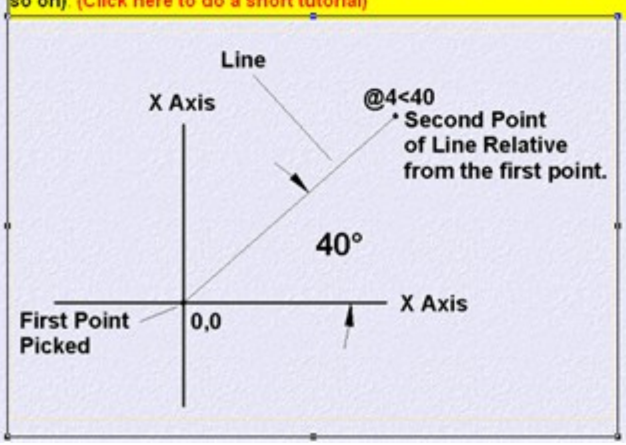
Relative Coordinate Entry

1. Type in: **LINE** (press enter)
2. Specify first point: Pick a point with your input device anywheres in the drawing area.
3. Specify next point Type in: **@2,0** (press enter) draws a line from the first point to the second point.
4. Specify next point Type in: **@0,2** (press enter) draws a line from the second point to the third point.
5. Specify next point Type in: **@-2,0** (press enter) Draw a line from the third point to the forth point in the negative 2 direction.

Remember: In relative coordinate entry each time a new point is picked that point then becomes the 0 reference point.

Polar Coordinate Entry

You would use polar coordinate entry if you know that you want to draw a line a certain distance at a particular angle. Lets say you wanted to draw a line 2 inches at 30 degrees, the polar coordinate entry would look like this @2<30, the @ symbol tells AutoCAD the next point is relative from the last point picked, the 2 tells AutoCAD the line will be 2 inches long, the < tells AutoCAD the next entry will be a degrees, the 30 tells AutoCAD the line will be drawn at 30 degrees. You must use polar coordinate entry in this manner or it will not work. **(Examples of polar coordinate entry: @1<90 , @4<30, @2<207 and so on)**. **(Click here to do a short tutorial)**



Polar Coordinate Entry

1. Type in: **LINE** (press enter)
2. Specify first point: Pick a point anywheres in the drawing area.
3. Specify next point type in: **@2<30** (press enter) Draws a line relative from the first point to the second, 2 inches in the 30 degree direction.
4. Specify next point type in: **@2<150** (press enter) draws a line relative from the second point to the third point, 2 inches in the 150 degree direction.
4. Type in: **CLOSE** (press enter) draws a line from the third point back to the first point to close the profile.

Osnap

When drawing in AutoCAD you should always try to draw as accurately as possible to do so Object Snap must be used. Object Snap is one of the useful tools found in AutoCAD. It increases accuracy, ability, performance, and productivity. The term object snap refers to the cursor ability to "snap exactly to a specific point or place on an object. For example, suppose you want to place a circle at the intersection of two lines. Normally you would try to pick the intersection, but would probably miss. Using object snap intersection you could snap precisely to the intersection of the two lines. When placing dimensions on a drawing you would most likely use one or two object snaps in combination with each other to do so. So the moral of the story is use object snaps when ever possible when drawing or placing dimension on an object. Object snaps can be turned on in groups using the drafting settings dialog. This is called running object snaps. For example before you start drawing you know you will be drawing at intersection, to endpoint, to center of circles and so on, you can turn them all on at one time. When ever you pass theses point's on an object the object snap for that point will automatically turn on. To turn osnaps on and off use the osnap button in the status bar at the bottom of the screen. When the button push in it is turned on, when it is out it is turned off.

Osnap **ENPOINT** draws a line form the endpoint of a line to another end point of a line.

Osnap **INTERSECTION** snaps to the intersection of two intersecting lines.

Osnap **NEAREST** snaps nearest to any point you pick on a line

Osnap **MIDPOINT** snaps to the exact midpoint of a line.

Osnap **TANGENT** snaps tangent from a circle or arc to another circle or arc.

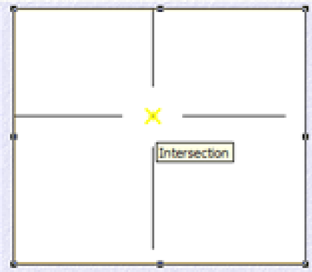
Osnap **CENTER** snaps to the exact center of a circle or arc.

USING OSNAPS INDIVIDUALLY

Osnap Intersection

How to draw using osnap Intersection

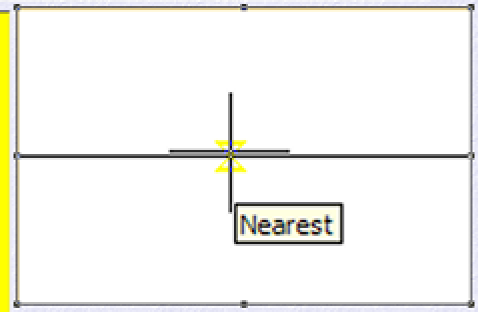
1. Draw 2 overlapping lines (see image)
2. Click on the **LINE** icon in the draw toolbar.
3. Specify first point: Move your cursor over to the intersection of the intersection line and click down when you see the yellow osnap intersection X at there intersection.
4. Move your cursor out a short distance and click and click down to end the line.
5. Press the **ESC** key on the keyboard to cancel.



Osnap Nearest

How to draw using osnap Nearest

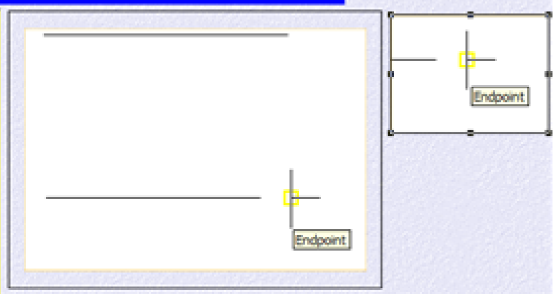
1. Draw a horizontal line any distance long.
2. Click on the **LINE** icon.
3. Specify first point: Move your cursor over the the horizontal line until you see the yellow hour class. You can click down anywheres along that line where you see the yellow hour glass that is the osnap nearest. Click down at any convent location.
4. Move your cursor up a short distance and click down.
5. Press the **ESC** key on the keyboard to end the line.



Osnap Endpoint

How to draw using osnap endpoint.

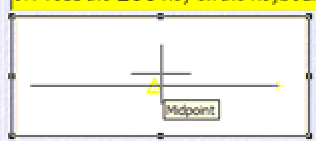
1. Draw two horizontal lines a short distance away from each other.
2. Click on the **LINE** icon in the draw toolbar.
3. Specify first point: Move your cursor to the lower right endpoint of the bottom line and click down when you see the yellow osnap endpoint box with the tooltip that said's endpoint.
4. Move your cursor up to the endpoint of the line above and click down when you see the yellow osnap endpoint box.
5. Press the **ESC** key on the keyboard to cancel.



Osnap Midpoint

How to draw using osnap midpoint

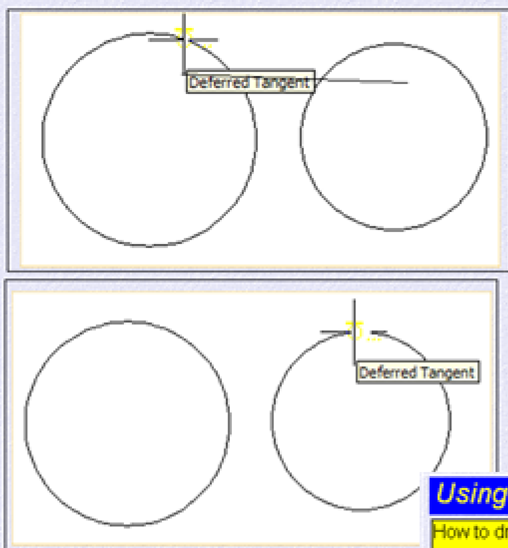
1. Draw a horizontal line any length
2. Click on the **LINE** icon in the draw toolbar.
3. Specify first point: Move your cursor along the horizontal line until you see the yellow osnap midpoint triangle then click down.
4. Move your cursor up a short distance and click down to end the line.
5. Press the **ESC** key on the keyboard cancel



Osnap Tangent

How to draw using osnap Tangent

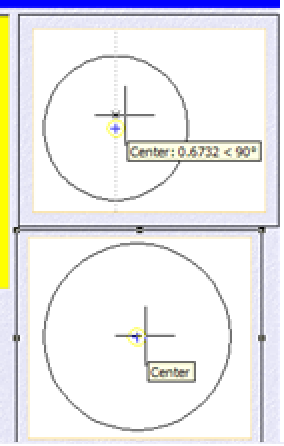
1. Draw 2 circle any diameter.
2. Click on the **LINE** icon.
3. Specify first point Type in: **TAN** (press enter) Move your cursor over to the circle on the right (on the line) until you see the yellow osnap **DEFERRED TANGENT** point then click down.
4. Specify second point Type in: **TAN** (press enter) Move your cursor to the circle on the other side (on the line) until you see the yellow osnap **DEFERRED TANGENT** point then click down. A line should then drawn tangent between the two circles.
5. Press the **ESC** key on the keyboard to end the line.



Osnap Center

How to draw using osnap Center

1. Draw a circle with any diameter
2. Click on the **LINE** icon in the draw toolbar.
3. Specify first point: Move your cursor to the center of the circle you created step in 1. Click down when you see the small yellow osnap center circle at its center.
4. Move your cursor out a short distance and click down to end the line.
5. Press the **ESC** key on the keyboard to cancel.

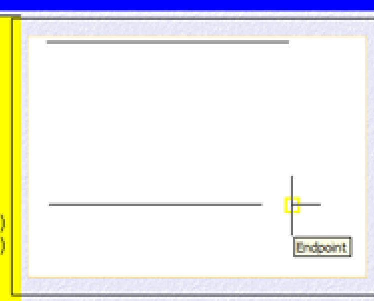


Using Osnap Individually

How to draw using osnap's individually

1. Draw two horizontal lines an 1" or 2" apart.
2. Turn off running object using the status bar **OSNAP** button, it should be out.
3. Click on the line icon in the draw toolbar.
4. Line specify first point: Type in: END (press enter)
5. Click on the bottom line on the right end.
6. Specify next point: Type in: **END** (press enter)
7. Specify next point: Type in: **END** (press enter)
8. Click on the top line on the right end.
9. Press the **ESC** key to cancel the command.

Any osnap command can be use individually by typing it in at the command. When AutoCAD ask you to pick a point in a drawing, think about what osnap to use for that situation.



Qnew/Open/save/Exit

Here you are given some of the ways of opening and closing an AutoCAD drawing Secession. How to start a new drawing from scratch. Also how to save a drawing to a drive or directory of your choosing.

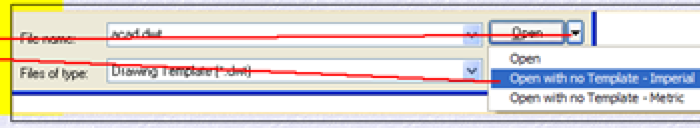
Qnew - Starts a brand new drawing from scratch.
Open - Used to open an already existing drawing.
Save - Save a drawing to a drive and directory of your choice for later use.
Exit - Used to exit out and close AutoCAD

Qnew Command

The Qnew command is used to quickly create a new drawing. By clicking on this icon the select template dialog box opens, from there you may create a new drawing from scratch "A blank piece of paper". This is the fastest way of starting a new drawing.

OPEN WITH NO TEMPLATE - IMPERIAL sets AutoCAD up to draw in decimal inch. If you wish to draw in metric units click on OPEN WITH NO TEMPLATE-METRIC

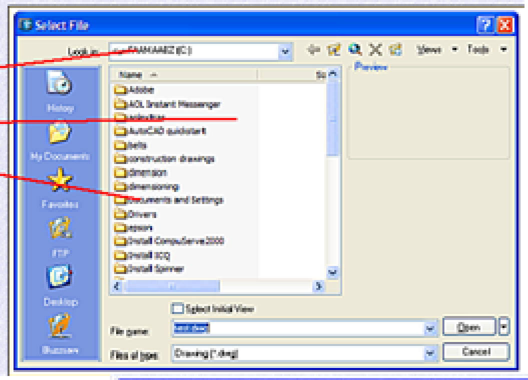
1. Click on the **QNEW** icon at the upper left corner of your screen in the standard toolbar.
2. Click on the **ARROW Down**.
3. Click on **OPEN WITH NO TEMPLATE - IMPERIAL**



Open Command

The open command is used to open an existing drawing already saved from a previous drawing secession.

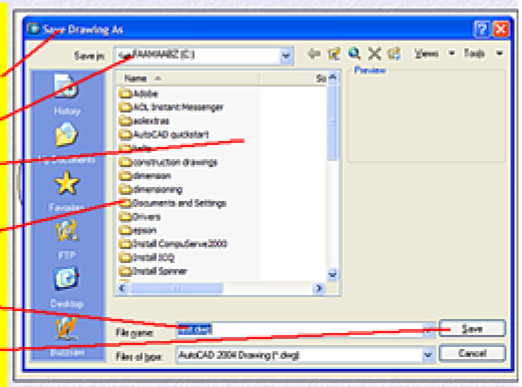
1. Click on the **OPEN** icon in the upper left corner of your screen in the standard toolbar. The **SELECT FILE** dialog box will open
2. Click here next to **LOOK IN** and you will see a drop down list of drives on your computer such as **A:,B:,C:D:** and so on.
3. Click on the drive your drawing is located on.
4. Scroll down the list of directories using the scroll bars to the right, find the directory your drawing is in and double click on that directory.
5. Once you have located your drawing you wish to open double click on it and it will open.



Save Command

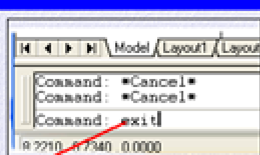
The save command is used to save the current open drawing. Use this command to save your drawing when you are ready to close your current drawing secession or start a new one.

1. Start a new drawing from scratch
2. Draw a circle and a line any length any diameter
3. Click on the **SAVE** icon in the upper left corner of your screen in the standard toolbar. The **SAVE DRAWING AS** dialog box opens.
4. Click here next to **SAVE IN** scroll down the list of drives on your computer and click on the drive you wish to save your drawing on. **(Example: A:,B:,C:,D:, etc).**
   NOTE: Once you have click on the drive you wish to save your drawing on, you may get a list of directory's you can click on a specified directory of your choice to save your drawing in.
   REMEMBER WHERE YOU SAVED YOU DRAWING
5. Click on this area next to **FILE NAME** then type in a name for your drawing **(Example: TEST)**
6. Click on **SAVE,** the drawing file will then be saved to the drive and directory that you chose in **step 4**.

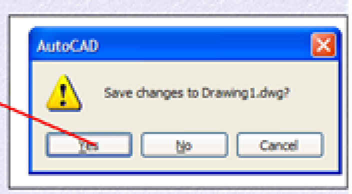


Exit Command

This is the best method to use when you want to exit AutoCAD. If any changes have been made to the drawing since you last saved it, exit invokes a warning box asking if you want to save changes to
the current open drawing before ending



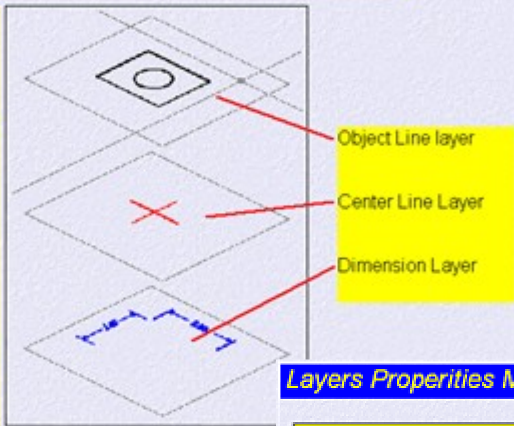
1. Type in at the command line: **EXIT** (then press enter)
2. A warning sign may or may not pop-up depending if you made any changes to your drawing since the last time you save it. If a warning sign does pop-up make you click on **YES** to save your changes. If you save yes to save changes the changes will be saved back the drawing that is currently open.



Layer

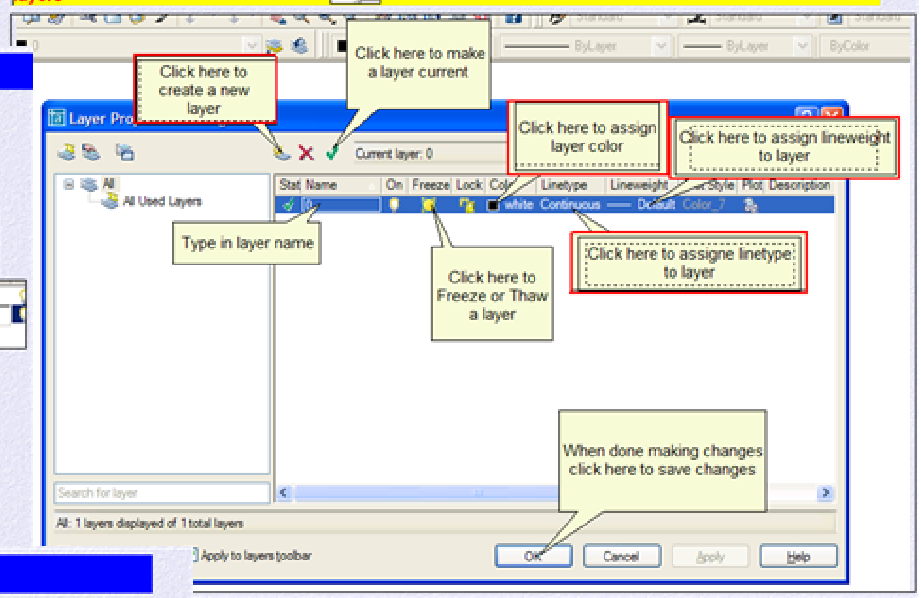
In a cad drawing, layers are used to group related objects in a drawing for viewing purposes or editing purposes. For example in a mechanical drawing you could have object lines on a layer named object or dimension lines on a layer name dimensions and phantom lines on a layer name phantom. In an architectural drawing you could have the floor plan on floor plan layer and the electrical on a layer named electrical and so on. Layers can be turned on or off to view only certain layers, they can be assigned different colors, linetypes, and lineweights. Layers are used mostly for viewing purposes. Just image you car, each major area of your car is on a different layer, your car body is on body layer, your chassis is on chassis layer, your engine is on engine layer. You turn off body layer, you can now see your engine and chassis without the car body obstructing your view. There can be an unlimited number of layers in an AutoCAD drawing.

Layer Properties Manager Dialog Box



Layers Properities Manager

This dailog box is used to control your layers turn on, turn off, make them current when ready to draw on, etc. To open this dialog box click on the layer icon in the layers toolbar "upper left on your screen". **How to use layers**



Create New Layers

Use this procedure when you want to create new layers

1. Click on the **LAYER** icon in the upper left corner of your screen
2. Click on the **NEW** button
3. Type in the **LAYER NAME** (Press enter on the keyboard)



Repeat step 2 and 3 if you need to create more than one layer. If you desire to leave the layer control box after you have created you layers don't forget to click on OK when you leave this will save your layers you have just created or else you will lose them.

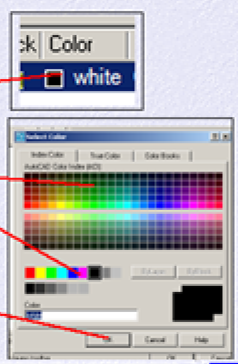
Layer 0 is the default AutoCAD layer it is always created when starting a new drawing.

Set Layer Color

This procedure is used to assigned a color to a layer. Once you assign a color to a layer everything created on that layer will take on the color assigned to that layer.

1. Under the color heading who's layer's color you wish to change click on the white icon.
2. Select a **COLOR** from the color palette, you may select any color you wish to assign to that layer.
3. Click on **OK** to close and save.

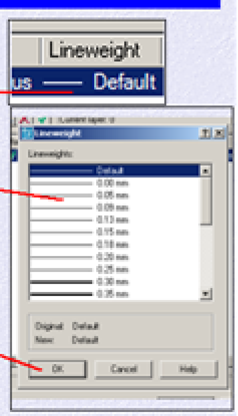
White is the AutoCAD layer default layer color it is created everytime a new layer is created.



Set Lineweight

This procedure is used to assign a line thickness to the linetype you assigned to that layer.

1. Under the lineweight heading who's layer's lineweight you wish to change, click on the **DEFAULT**.
2. Scroll down the list of **LINEWEIGHTS** and click on the desired lineweight to assign to the linetype assigned to that layer.
3. Click on **OK** to close and save



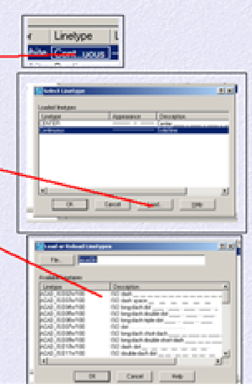
The default lineweight is the AutoCAD default lineweight it is created everytime a new layer is created.
To have the lineweight display on your screen click on the LWT button in the status bar at the bottom of your screen. When the button is in the in position that option is turned on.

Set Linetype

Use this procedure when you want to assign a particular linetype to a layer for use in your drawing.

1. Under the linetype heading who's layer's linetype you wish to change, click on **CONTINUOUS**.
2. The **SELECT LINETYPE** dialog box will appear, click on **LOAD.**
3. The **LOAD AND RELOAD LINETYPES** dialog box will appear. Scroll down the list of linetypes and double click on the desired linetype you wish to load.
4. The **SELECT LINETYPE** dialog box will reappear, click on the linetype you selected in step 3, then click on **OK** that linetype them becomes assigned to that layer.

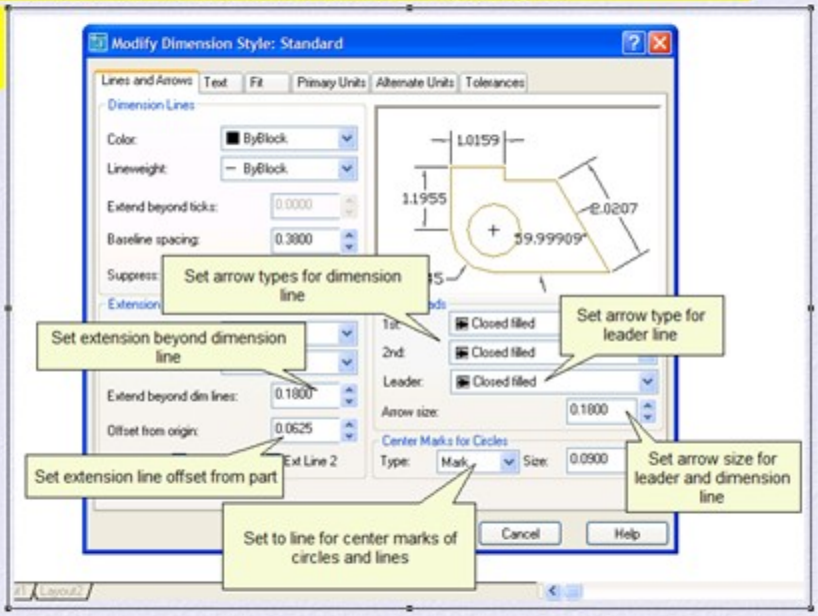
Continuous linetype is the AutoCAD default layer linetype it is created everytime you create a new layer.



Dimensioning

To make dimension setting in preparation for dimensioning a drawing click on the STYLE… Option in the dimension pulldown menu at the top of the screen. Then click on the MODIFY button to open the dimension style dialog box.

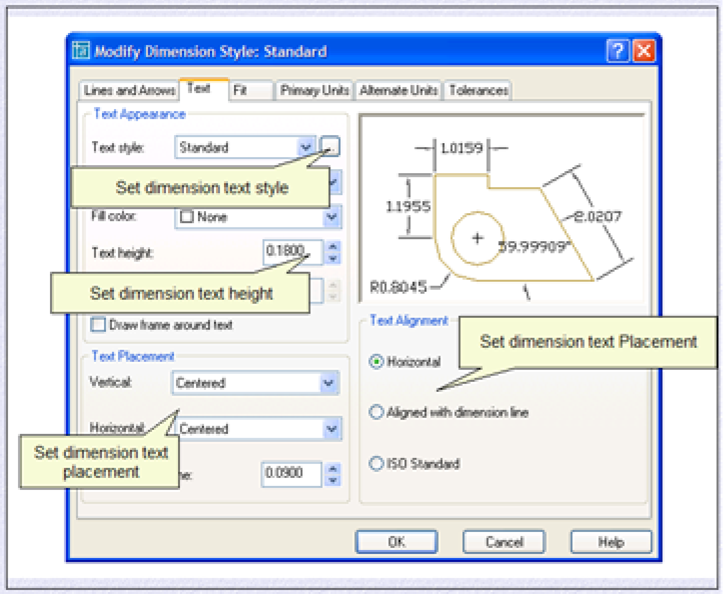
Set lines and arrows
Set dimension text style and height
Set dimension units
Using dimensions



Dimensioning

To make dimension setting in preparation for dimensioning a drawing click on the STYLE… Option in the dimension pulldown menu at the top of the screen. Then click on the MODIFY button to open the dimension style dialog box.

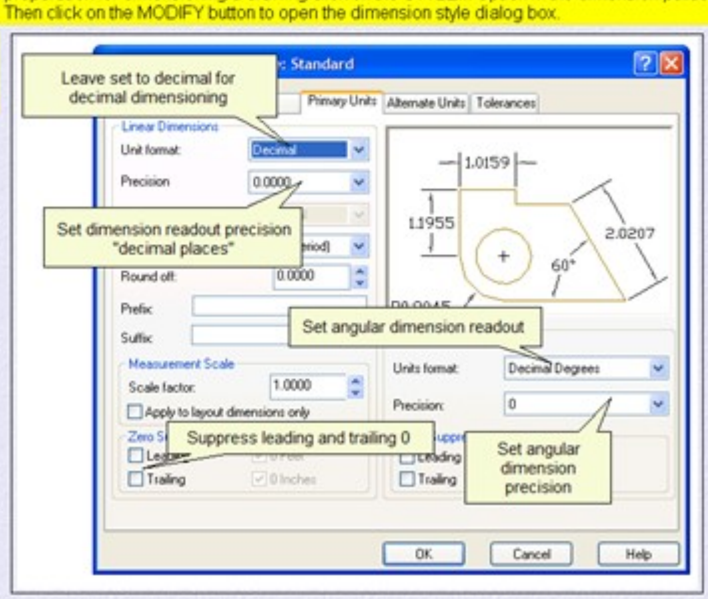
Set lines and arrows
Set dimension text style and height
Set dimension units
Using dimensions



Dimensioning

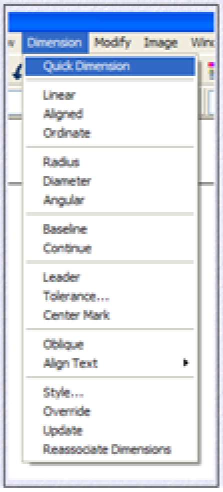
To make dimension setting in preparation for dimensioning a drawing click on the STYLE… Option in the dimension pulldown menu at the top of the screen. Then click on the MODIFY button to open the dimension style dialog box.

Set lines and arrows
Set dimension text style and height
Set dimension units
Using dimensions



Using Dimensions

After you have made your dimension settings in the dimension style dialog box and you are now ready to dimension your drawing click on the DIMENSION pulldown menu scroll down the list of dimension commands and click on the dimension command you wish to use. Click on the OSNAP button in the status bar at the bottom of your screen so you can dimension with accuracy.



Linear - Specify by picking two points on an object then pick a location for dimension.
Aligned - Specify by picking two points on angle line then pick location for dimension.
Radius - Specify by picking a radius on an object then picking leader location.
Diameter - Specify by picking a circle then picking leader location
Angular - Specify by picking two lines then pick location of dimension
Leader - Specify by picking on the object you want the leader to point at then pick leader location.
Center Mark - Specify by picking on circle or radius to apply center marks to.

The above are the basic dimension commands there are more but these are the basics.
If you have turned on the OSNAP on the status bar at the bottom of your screen turn it off now, it may cause some confusion.
When using dimension commands always watch your command for further instruction "Always".
Editing Dimensioning Text, **Editing Dimensions with grips**, **Changing Dimension Settings**.

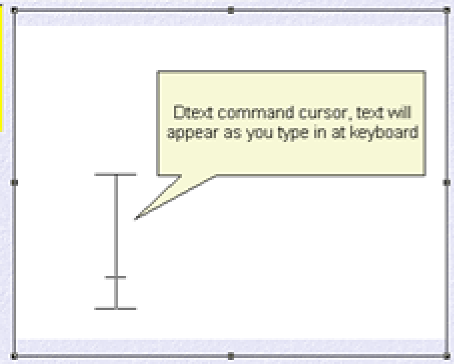
Text In A Drawing

In this book you learn one way of putting text into a drawing "Single line text" and how to edit it. This is only one way there are others, this is a good start for a beginner.

Dtext
DDedit

Detext Command

The Dtext command places individual lines of text in a drawing and allows you to see each letter as it is typed. You may enter multiple lines of text without exiting the dtext command by pressing the enter key when you get to the end of one line, it will then rap to the next line. Pressing the spacebar on the keyboard will put a spacing between letters and words.

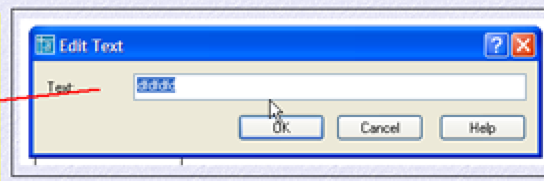


1. Type in: **DTEXT** the press **ENTER** on the keyboard.
2. Specify start point of text: Click anywheres in your drawing area where you want your text to start, pick with the mouse.
3. Specify height: Type in the height that you want your text to be then press **ENTER** on the keyboard.
4. Specify rotation angle of text Type in: **0** (press enter). This sets the rotation of the text.
5. Enter text: Type in the text you desire then press **ENTER** on the keyboard. If you desire more than one line of text press enter again to type in another line of text.
6. When you are done typing in the desire text press **ENTER** on the keyboard twice.
   NOTE: If you wish to enter text in another area of your drawing without exiting the dtext command, before you do step 6 click in another area of the drawing with your mouse then start typing in again, when done do step 6.

DDedit Command

The Ddedit command invokes a dialog box for editing single line text. It only allows you to edit the characters, not the height or style. It is a useful
command when you want to edit missed spelled word in a single line text.

1. Type in: **DDedit** (Press enter)
2. Select an annotation object: with the mouse select the line of text you wish to edit.
3. The **EDIT TEXT** box will open displaying the text you selected in step 2. Using the keyboard edit the text in the box. When done editing click on **OK** to save changes. The text will now update in the drawing.
   You may select another line of text at this or press the ESC key to cancel the command.



Zoom Commands

To zoom means to "magnify" make larger or small a segment or area of a drawing. To "zoom out" means to display a larger area of a drawing. To "zoom in" means to make a smaller area of a drawing larger as to see more detail. Zooming does not change the size of the drawing objects, zooming only changes the display of the objects. Zooming is like holding an object in your hand, you move the object closer to your eyes you are "zooming in" to see more detail, if you move the object away from your eyes you are "zooming out" you see less detail but you see more of the object in its entirety.

Zoom Realtime
Zoom Window
Zoom Previous
Zoom All
Pan Realtime

Zoom Window

The "zoom window" command is used to zoom in on a specific area in your drawing by opening up a window around the specified windowed area or object.

1. Draw a circle and a line any diameter any length
2. Click on the **ZOOM WINDOW** icon at the top of the screen in the standard toolbar menu area.
4. Specify first corner: **Pick a point to the left and just below the line.**
5. Specify opposite corner: **Pick a point just above and to the right of the line.**

You have zoomed on just the line by specifying a zoom with a window option. This can be done anywheres within the drawing area.

STOP HERE click on the **BACK button** on this page to go back to the **STANDARD TOOLBAR** page and click on the **ZOOM PREVIOUS** icon.

Zoom Realtime

The "zoom realtime" command is used when you would like to zoom in on a specific location in you drawing. You can also use this zoom realtime to

1. Draw a circle and a line any diameter any length
2. Click on the **ZOOM REALTIME** icon located in at the top of the screen in the standard toolbar menu area. Move your cursor down into the drawing area your cursor will change into Magnifying glass with a + at the top and a - at the bottom.
3. Click down with the left mouse button and hold it down move your cursor in the direction of the + you will get closer to the objects "you are zooming in". Move your cursor in the direction of the - you will go further away "you are zoom out" from the objects.

Press the **ESC** key on the keyboard when in the desired zoom magnification

Zoom Previous

The "zoom previous" command is used to zoom out to the previously zoomed area were you were at before you zoomed in. This command can be used to zoom all the way back out to the beginning of your first

This command is used in conjunction with any of the other zoom commands.

1. Create some objects in the drawing area anything will do. Zoom in using one of the other zoom commands.
2. Click on the **ZOOM PREVIOUS** icon at the top of the screen in the standard toolbar menu are.

Zoom All

The "zoom all" command when used, displays all object within the drawing area.

1. Draw so objects on the screen, some lines ect.
2. Zoom in on the objects using the zoom window option of the zoom command
3. Type in ZOOM then press enter
4. Type in ALL then press enter

Pan Realtime

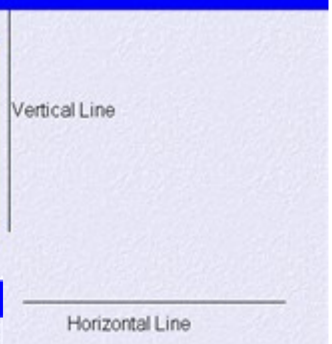
The "pan realtime" command is used when you would like to pan around in your drawing to look at different objects inside your drawing area. You can pan right, left, up, and down in any direction you like to see any part of your drawing. Remember the drawing itself is not moving, the viewer "you" are moving not the objects in the drawing.

1. Draw a circle and a line any diameter any length
2. Click on the **PAN REALTIME** icon at the top of your screen in the standard toolbar menu area. Move your cursor down into the drawing area your cursor will change into a hand.
3. Click down with the left mouse button and hold it down, then move your cursor around in the drawing area. The objects seem to be moving but that is not true you are moving " the viewer" not the objects.

Press the **ESC** key on the keyboard when in the desired pan location.

Ortho

The ORTHO command allows you to draw straight horizontal and vertical line while using the line command. The way to turn it on is in the status bar at the bottom of your screen. When the ortho button is in ortho is turned on.



ESC Key

The ESC key on the keyboard is cancel command in autocad. When ever you get lost and confused "No not the song" it is best to just cancel the command you are in and start it over again.

How Commands Work

There are basicly three ways to entering commands into autocad - pulldown menus, toolbars, typing them in at the command line. For the beginner it is good to use either the pulldown menus or the toolbars to enter commands. When you envoke a command from a pulldown menu or toolbar watch the command line for further instructions or question autocad may have to ask you in order to complete the command or task you are trying to do. For example you select the line command from the draw toolbar at the command line autocad will ask you to select first point. Each command is different so always watch the "command line" for further instructions.

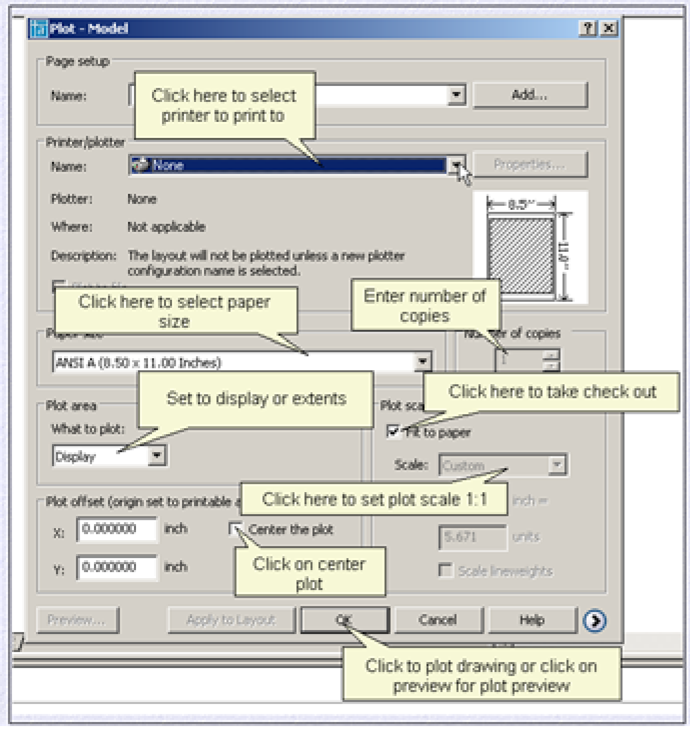
Plot Command

In AutoCAD the term "Plot" refers to plotting on a plot device such as (inkjet or laser plotter) or printing on a laserjet or inkjet printer. The plot command is used to invoke the plot dialog box where all the plot or print settings are made before you send the drawing to the printer or plotter. A printer or plotter must be connected to your computer. Each plotter or printer setup is somewhat different, you may have to experiment with the basic setup before getting it right. To access the plot dialog box to plot

1. Click on **FILE** pulldown menu
2. Click on **PLOT**

To get a preview of what will print out before it is printed, click on **PREVIEW**. If everything looks good, right click with mouse then click on **PLOT** or click on **EXIT** to go back to plot dialog box to adjust settings

Plot and print has the same meaning in AutoCAD



Typical Drawing Setup

Once you have started a new drawing from scratch it is a good thing to make the following settings before you begin to draw:

1. Set **GRID** (optional)
2. Set **SNAP** (optional)
3. Set **UNITS**
4. Set **LIMITS**
5. Do a **ZOOM** with the **ALL** option
6. Create **LAYERS** assign **LINETYPES, LAYER COLOR,** and **LINEWEIGHTS**. Read layers section.
7. Set **LTSCALE**
8. Set **TEXT STYLE**
9. Turn on **LWT**
10. Draw a Border and Titleblock using the limits settings as a guideline.



Setting Up Grips

The grid commands places a pattern of dots on the screen at any spacing. This command is used as a drafting aide

1. Type in: **GRID** (Press enter on the keyboard)
2. Type in: **.25** (Press enter on the keyboard)

You may set the grid spacing to any size you wish. Start out with a .25 spacing then adjust as needed. You can use the GRID button in the status bar to turn on and off grid

Setting Snap

When you move your pointing devices, the cursor crosshairs move freely on the screen. Sometimes it is hard to place a point accurately. The snap commands allows the cursor to move only in exact increments.

1. Type in: **SNAP** (Press enter on the keyboard)
2. Type in: **.25** (Press enter on the keyboard)

You may set the Snap spacing to any size you wish. Start out with a .25 spacing then adjust as needed. It is a good idea to keep the grid spacing and snap spacing the same to avoid confusing. You can use the SNAP button in the status bar to turn on and off snap.

Zoom All

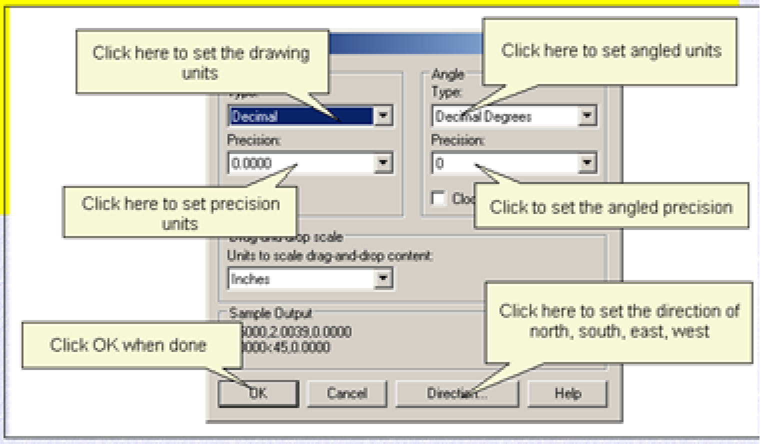
The "zoom all" command when used, displays all object within the drawing area.

1. Draw so objects on the screen, some lines ect.
2. Zoom in on the objects using the zoom window option of the zoom command
3. Type in ZOOM then press enter
4. Type in ALL then press enter

Units Command

When drawing in autocad you have the option of drawing in many different types of drawing units such as decimal units for mechanical drawings and architectural units for architectural drawings. For drawing in metric units the scale factor is 25.4. So here's what you do when starting a new drawing and you want to draw it in metric. There is no metric settings in autocad you have to convert all settings using the 25.4 scale factor "Everything". For example the limits command the lower left corner is 0,0, but the upper right corner is 17,11. "B" size sheet. You multiple the 17 by 25.4 then you multiple the 11 by 25.4 this gives you the limits metric equivalent to a "B" size sheet. "Do this for every setting you make" . Once you make all settings for metric you can using
distances. For example when using the line command a distance of 50mm is entered as 50.

1. Type in: **UNITS** (Press enter on the keyboard)
2. Under **TYPE** click on **DECIMAL**
   If you wish Architectural units click on Architectural.
3. Under **PRECISION** click on **0.000**
   You may set the precision to any precision you wish. Start out with 2 or 3 decimal places then adjust as needed.
4. Click on **OK** to close and save settings.



Setting Limits

The limits command is used to set the area to drawn in. For example you have chosen a "B" size sheet of paper to draw on. The dimension for a "B" size sheet of paper are 11 X 17, so then you would set your limits to 0,0 for lower left corner, for upper right corner you set to 17,11 (just the reverse of 11X17). Remember autocad always reads X direction first and Y direction second. Use the UCS icon at the lower left corner as a reference. After you have set your limits to the paper size your going to use to draw on, draw the outline of the paper using the line command.

1. Type in: **LIMITS** (Press enter on the keyboard)
2. For **LOWER LEFT CORNER** type in: **0,0** (Press enter on the keyboard)
3. For **UPPER RIGHT CORNER** type in the dimension of your sheet size (Example **17,11** for B size then press enter on the keyboard)

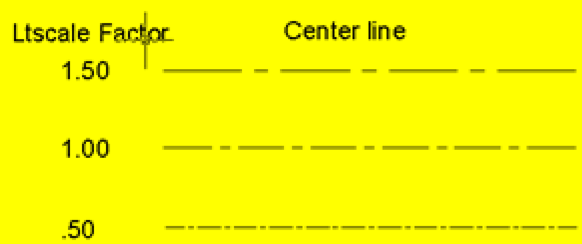
For sheets sizes of A,B,C,D,E set the upper right corner to:

A size = 11,8.5
B size = 17,11
C size = 22,17
D size = 36,24
E size = 44,36

Leave the lower left corner to 0,0 (always)

Ltscale

The ltscale (lintype scale) set the length of dashes in linetypes having them. The ltscale command works only on lines with dashes and spaces in them such as hidden, dashed, phantom, center, etc.



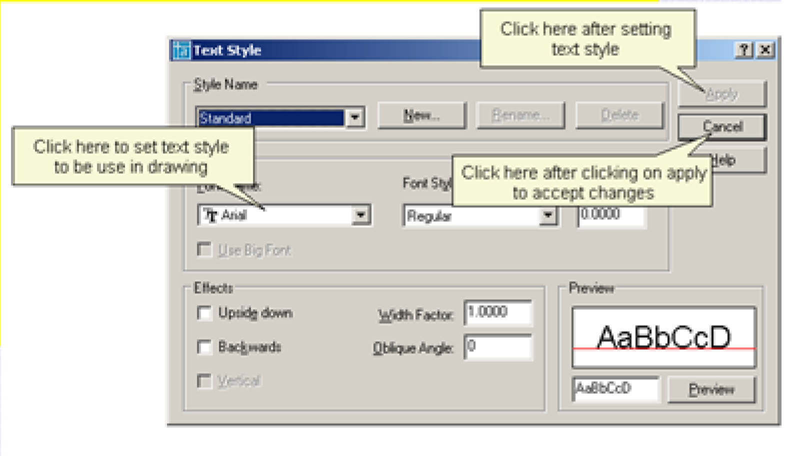
1. Type in: **LTSCALE** (press enter on the keyboard)
2. Type in: **.7** (press enter on the keyboard)

Play with this command to get feel on how autocad determines line scale factors.

Set Text Style

The style command is used to set a font or text style to be used in a drawing

1. Type in: **STYLE** (press enter on the keyboard)
2. Scroll down the list until you find the desired **TEXT STYLE** and click on it. After you have click on the desired text style the **APPLY** button will then turn white.
3. Click on **APPLY**
4. When you click on apply the cancel button will change into a **CLOSE** button. Click on **CLOSE** to save the text style to be used in your drawing.



Putting It All Together

"Putting it all together" The logical sequence of things to do when starting a new drawing. If you are opening an already existing drawing from a previous drawing session then there is know need to do the following.

1. Start a new drawing using the QNEW command
2. Go thru the "Typical Drawing Setup" section in this book
3. Make dimension settings thru the dimension style manager dialog box
4. Draw a border and titleblock if you haven't already
5. You are now ready to start the drawing process. The first things you start to draw in autocad are usually lines or in drafting terms they are called "Object Lines" so you should have a layer called "Object Lines" created, if not create it now. Make the object layer current. Once you have made the object layer current go ahead and start drawing object object lines using the line command. (after you have make "Object Lines" your current layer make sure you click on OK before leaving the layer control dialog box).
6. After you have drawn your object put dimensions and text in there places.
7. Plot your drawing or save it for later

Turn On LWT

The lwt (lineweight) button in the status bar at the bottom of the screen is used to show lineweights that were assigned to layers in the layer control manager dialog box, on the screen. For example you created a layer call object layer and assigned it a continuous linetype, and a lineweight of 35mm. In order for you to see the lineweight on your screen you would have to turn on the LWT button in the status bar at the bottom of the screen.

1. Click on the **LWT** button in the status bar at the bottom of the screen

Toolbars

Toolbars are used to easily access autocad commands. Toolbars can be turned on or off by doing the following. Right-click any toolbar and click a toolbar on the shortcut menu.



1. Move your cursor over any toolbar and right click your right mouse button.
2. Move down the menu and click on the toolbar you wish to open, it then appears in the drawing area.

NOTE: The standard toolbars that are open when you first open AutoCAD are: DRAW, LAYERS, MODIFY, PROPERTIES, STANDARD, STYLES, INSERT.

When the toolbar appears in the drawing area move your cursor over the blue bar in the toolbar, click and hold down the left mouse button. Move the toolbar to a location at the edge of the screen and let up on the mouse button. This is called docking the toolbar.

Undo/Redo

The undo command undoes the last command you did If you just drew a line and wanted to undo it, click on the undo icon:

1. Draw a line any length in the drawing area using the line command.
2. Click on the undo icon at the top of the screen, the line is no more. (Go onto redo)



The redo command undoes an undo. For example you just invoked the undo command to undo a line you just had drawn and now you decided you wanted that line after all, click on the redo command to get it back again.

1. Click on the redo icon at the top of the screen to bring back the line you undone using the undo command. Presto the line is back.

Keyboard Function Keys

The ESC key and the F8 keys have special functions in Autocad.

ESC - This is the command cancel key. When ever your in a command and just need to start over press this key, it cancels the current command you are in.

F1 - Opens the AutoCAD help dialog box.

F2 - Opens the AutoCAD text window. In this window you can scroll through the history of commands.

F3 - Turns running osnap on or off

F4 - Turns tablet on or off (You will probably never use this key)

F5 - Switches between ISOPLANES (top,front,right)

F6 - Turns coordinate readout at lower left of screen on or off

F7 - Turns Grid display on or off

F8 - Turns ORTHO on or off (ortho is used in conjunction with the line to draw straight horizontal and vertical lines.

F9 - Turns SNAP on or off.